fe-safe™ component for Isight

Integrating Design Durability in the Simulation Process Flow by Safe Technology

User Manual

Contents

Disclaimer Notice

This material is for use only by the recipient of this material. A recipient or other user (a "licensee") is not permitted to copy this document.

The information in this document is subject to change without notice and should not be construed as a description of any software supplied by or on behalf of Safe Technology Limited (STL) or as any commitment by STL.

STL assumes no responsibility for any errors that appear in this document, the software to which it pertains or in any material provided as part of a technical support service for the Software.

The software described in this document is furnished under licence and may be used only in accordance with the terms of such licence. The Software and the instructions for its use are provided "as is" without warranty of any kind. STL and, as appropriate, any reseller further disclaim all implied warranties including without limitation any implied warranties of merchantability or of fitness for a particular purpose. The entire risk arising out of the use or performance of the Software and documentation remains with the licensee.

In no event will STL, its authors, or anyone else involved in the creation, production, sale or delivery of the Software including, as appropriate, any reseller be liable for any damages whatsoever and howsoever arising (including, but not limited to, damages for loss of business profits, business interruption, loss of business information, or other pecuniary loss) arising out of the use or inability to use the Software or documentation or a fault in the same, even if STL or the reseller has been advised of the possibility of such damages.

STL accepts liability only to the extent as may be agreed and expressly set out in a licence to use the Software entered into with a licensee.

No part of this document may be reproduced in any form or distributed in any way without the prior written agreement of STL. Should the licensee wish to make any copy of this document, prior written permission must be sought from STL, Willis House, Peel Street, Sheffield, S10 2PQ, United Kingdom. Tel: +44 (0) 114 268 6444, Fax: +44 (0) 114 303 0055, Email: info@safetechnology.com

Copyright© 2011 Safe Technology Limited

Proprietary data.

Unauthorized use, distribution, or duplication is prohibited.

All rights Reserved.

Trademark and Registered Trademark Acknowledgements

- *fe-safe®*, *fe-safe/Composites®*, *fe-safe/TMF®*, *fe-safe/TURBOlife®*, *safe4fatigue®* and *safe4mat®* are registered trademarks of Safe Technology Limited or its subsidiaries in the United Kingdom and/or other countries.
- Verity® is a registered trademark of the Battelle Institute.
- Adobe, the Adobe logo and Acrobat are either registered trademarks or trademarks of Adobe Systems Incorporated in the United States and/or other countries.
- Microsoft, Windows, Windows XP, Windows Vista and Windows 7 are either registered trademarks or trademarks of Microsoft Corporation or its subsidiaries in the United States and/or other countries.
- Abaqus and SIMULIA are trademarks or registered trademarks of Dassault Systèmes or its subsidiaries in the United States and/or other countries.
- MSC., MSC Nastran, MD Nastran, Patran and MD Patran are trademarks or registered trademarks of MSC.Software Corporation or its subsidiaries in the United States and/or other countries.
- NX is a trademark or registered trademark of Siemens Product Lifecycle Management Software Inc. or its subsidiaries in the United States and in other countries.
- NASTRAN is a registered trademark of the National Aeronautics Space Administration.IDEAS Master-Series is a registered trademark of Structural Dynamics Research Corporation.
- ANSYS, ANSYS Workbench and any and all ANSYS, Inc. brand, product, service and feature names, logos and slogans are trademarks or registered trademarks of ANSYS, Inc. or its subsidiaries located in the United States or other countries.
- HyperWorks, HyperView, OptiStruct, RADIOSS, the Altair Engineering Logo and all Altair Engineering product names are trademarks or registered trademarks of Altair Engineering Inc., USA.
- Pro/ENGINEER and Pro/Mechanica are registered trademarks of Parametric Technology Corporation.
- µETA is a trademark of BETA CAE Systems S.A.
- Cadfix is a registered trademark of International TechneGroup Incorporated (ITI)
- FEMGV and FEMVIEW are registered trademarks of TNO DIANA BV.
- Linux® is a registered trademark of Linus Torvalds.
- FlexNet is a registered trademark of Flexera Software Inc..
- All other brand, product, service and feature names or trademarks are the property of their respective owners.

Copyright© 2011 Safe Technology Limited

1 Introduction to the *fe-safe* component for Isight

1.1 About *fe-safe*

fe-safe is a powerful, comprehensive and easy-to-use suite of fatigue analysis software for Finite Element models. It is used alongside commercial FEA software to calculate:

- where fatigue cracks will occur;
- when fatigue cracks will initiate;
- the factors of safety on working stresses (for rapid optimisation);
- the probability of survival at different service lives (the 'warranty claim' curve).

Results are presented as contour plots which can be plotted using standard FE viewers.

fe-safe has direct interfaces to the leading FEA suites.

1.2 About *Isight*

Isight provides a suite of visual and flexible tools to set up and manage computer software required to execute simulation-based design processes, including commercial CAD/CAE software, internally developed programs, and Excel spreadsheets. The open API supports the development of custom interfaces to link additional in-house and commercial applications by partners and customers.

1.3 About the *fe-safe* component for Isight

The *fe-safe* component for Isight is an added capability which allows for *fe-safe* to either be inserted into an existing simulation workflow or define its own, with subsequent analyses (parametric studies, DOE, optimizations etc.) being conducted from within Isight.

Users of the *fe-safe* component for Isight are assumed to have a working knowledge of *fe-safe*, including such techniques as configuring a fatigue analysis and setting properties for different parts of the model, defining the fatigue loading, running an analysis and exporting fatigue results. The use and application of *fe-safe* is described in the *fe-safe User Manual*, which should be referred to alongside the *Isight Getting Started Guide* from Simulia.

Features of the *fe-safe* component for Isight include:

- Support for Finite Element solutions from Abaqus ODB files.
- Parametric analysis of numerical parameters, which automatically enables DOE and Optimization processes to be defined.
- Manipulating string parameters externally (e.g. via *.txt files)

1.4 How to Use This Manual

Chapter 2 (Using the *fe-safe* component for Isight) provides an overview of the actions required in order to perform an *fe-safe* analysis as part of a simulation workflow in Isight. It also contains details on installing and publishing the *fe-safe* component in Isight and briefly explains how the component works. Chapter 3 (Interface Reference) offers further details of the available options. The prerequisites for FE models to be used with the *fe-safe* component for Isight are explained in chapter 4 (Preparing FE Models for use with the *fe-safe* component and Isight). Worked examples of the process are provided in tutorials A and B.

Users new to *fe-safe*

Because this manual assumes some familiarity with *fe-safe*, it will be necessary to learn a little about the main program first. Work through some of the tutorials in the *fe-safe User Manual*, including at least one demonstrating the use of data from Abaqus, then return here.

***fe-safe* users new to Isight**

Some understanding of the Isight workflow is required to make the best use of the software. The *Isight Getting Started Guide* from Simulia provides learning material for those unfamiliar with the software.

Read Chapter 4, Chapter 2, then work through Tutorials A and B to familiarise yourself with the *fe-safe* component in Isight. Once you have done this, follow the procedure described in Chapter 2 with your own data, referring to Chapter 3 as necessary.

Experienced users of Isight and *fe-safe*

Experienced users are most likely to refer to Chapters 2, 3 and Tutorial A as a working reference.

2 Using the *fe-safe* component for Isight

2.1 Installation Instructions

For convenience, the following path contraction is used in the instructions:

Shorthand	Typical Location
`<fe-safe-installation-dir>`	`C:/Program Files (x86)/Safe_Technology/fe-safe/version.6.x`

Find the directory `components` under the *fe-safe* installation directory. The provided zip file needs to be unzipped into this directory, which will then look like this:
`<fe-safe-installation-dir>/components/Isight/fe-safe.jar`

Open *Isight Design Gateway*, and publish the component by going to the **View** menu then selecting **Library**. In the new dialogue that opens, click the second icon  and navigate to the above file `fe-safe.jar`. Click **Publish**. You can then add the component icon to the main *Isight* palette by clicking **Add to Palette**.

***Note**: If you have received an updated version of the component jar file, you need to publish it again. After publishing you must close and restart* Isight Design Gateway *for the changes to take effect.*

The final step in setting up the component is to indicate to *Isight* where your *fe-safe* installation directory is; this only needs to be done once. Go to **Edit** then **Preferences**, and in the dialogue that opens up, expand the **Components** section in the left-hand tree. Click the **fe-safe** node, and in the panel on the right set your *fe-safe* installation directory (either by browsing or typing in a path name). This is typically `C:/Program Files (x86)/Safe_Technology/fe-safe/version.6.x`.

The component is now ready to use in an *Isight* model.

2.2 Understanding the component

The *fe-safe* component can be run as a single component within an *Isight* model, or linked to other components within the same model. A schematic is presented in Figure 2-1 below.

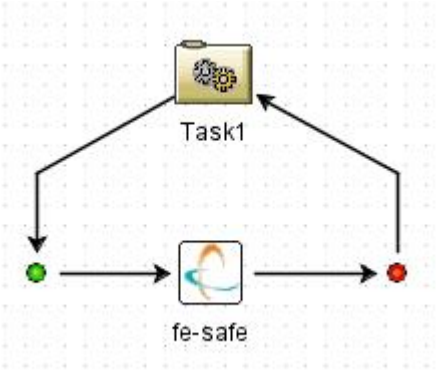

Figure 2-1: The *fe-safe* component inserted into an Isight workflow

The component's task is to define and run an *fe-safe* analysis. For this it needs an *fe-safe* project to be configured based on an FEA solution file which is either received from upstream systems or specified whilst setting up the cell. An *fe-safe* project must be defined for the component to operate correctly. You can either load a project that was previously setup with *fe-safe*, or start a brand new project from within *Isight*.

Once a suitable project is created or loaded, the *fe-safe* component reads the project settings and gives you the option to parameterise them within *Isight*. Then you can connect the parameters upstream and downstream of *fe-safe* as normal.

2.3 How to use the *fe-safe* component

2.3.1 Setting up the project

The very first stage in setting up an *fe-safe* component is to set up the project that defines the task the component will perform when executed. There are two ways of setting up the project:

- You can create the project in *Isight* by indicating one or more FEA solution files and a project directory for *fe-safe* to store its settings in,
- If you have previously set up an *fe-safe* project based around one or more FEA solution files and want to use *Isight* to perform a parametric study on this analysis, you can load this project by specifying the project directory.

These two options are explained in detail below.

How to create a project

Double-click the *fe-safe* component in the *Isight* model or right-click and select **Edit**. This opens up an editor dialogue shown in Figure 2-2:

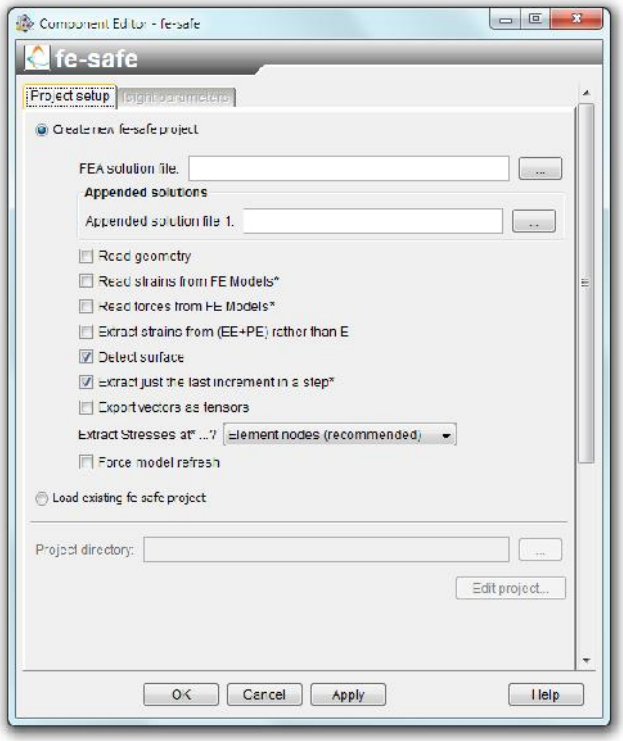

Figure 2-2: The *fe-safe* component editor

The **FEA solution file** box is used to indicate to *fe-safe* what FEA solution to open. The **Appended solution** box can be used to append additional FEA solution files. The selection boxes below control the data to be read from the solution files.

***Note**: At this release pre-scanning of FEA solutions is not supported when using* Isight*, as* fe-safe *has no way at present to replay the pre-scan choices when the solution is refreshed by subsequent runs. The FEA solution is loaded in full every time it is updated by components upstream of* fe-safe*.*

When you have indicated the primary FEA solution file, the **Project Directory** box is automatically populated with a path to a project directory, based on the name of the solution file. This can be changed to any chosen path as required. For portability reasons it is recommended to keep all the *Isight* model resources together in the same directory.

Now the solution file is known, you can configure the analysis. Click the **Edit project…** button at the lower right. This opens the *fe-safe* GUI in a component mode and starts to load the model in full. This is a special mode used when some other process is controlling the input and output files for *fe-safe*, and accordingly all controls that allow a user to manipulate the paths to those files have been removed. Certain other controls have been removed for features that are not available when using *fe-safe* as a component in a workflow (see Limitations). Otherwise the GUI is identical to the normal operation of *fe-safe*.

When you've finished setting up the project, simply close the GUI. The project is ready to be parameterised, see the Parameterisation section.

How to load a project

Select the **Load existing *fe-safe* project** option at the bottom of the dialogue. This enables the **Project Directory** box where the directory of the project you want to load must be selected. The project is ready to be parameterised, see the Parameterisation section.

If an output solution file was configured in the loaded project, the path will be ignored but the filename and file type will be respected. This is because *Isight* default behaviour is to write output files into its runtime directory. The default behaviour and configuration for the output file parameter can be changed in the *Isight Design Gateway* under the **Files** tab.

***Note**: Projects configured using the pre-scanner to read the FEA solution file may not load as expected. When the component executes, it will perform a full-read of the model and consequently the dataset numbers will change; which will most likely invalidate your analysis setup. If you're setting up a project with the intention of using it later within* Isight*, skip the pre-scan step and do a full-read instead.*

2.3.2 Parameterisation

After setting up the *fe-safe* project, certain project settings can be promoted to *Isight* parameters. Click on the **Isight parameters** tab in the editor as shown in Figure 2-3. This presents a hierarchy of all the settings that are available for parameterisation in the project. To promote a setting to an *Isight* parameter, find it in the tree and click **Promote** (and similarly click **Demote** to remove the selected parameter).

For example, to make "Young's Modulus" for Group 1 material available as a parameter, find it in the tree under Job > Material Databases > Materials > Material 1. Click **Promote**, and this setting appears in bold and in the second tree called **Promoted Settings**.

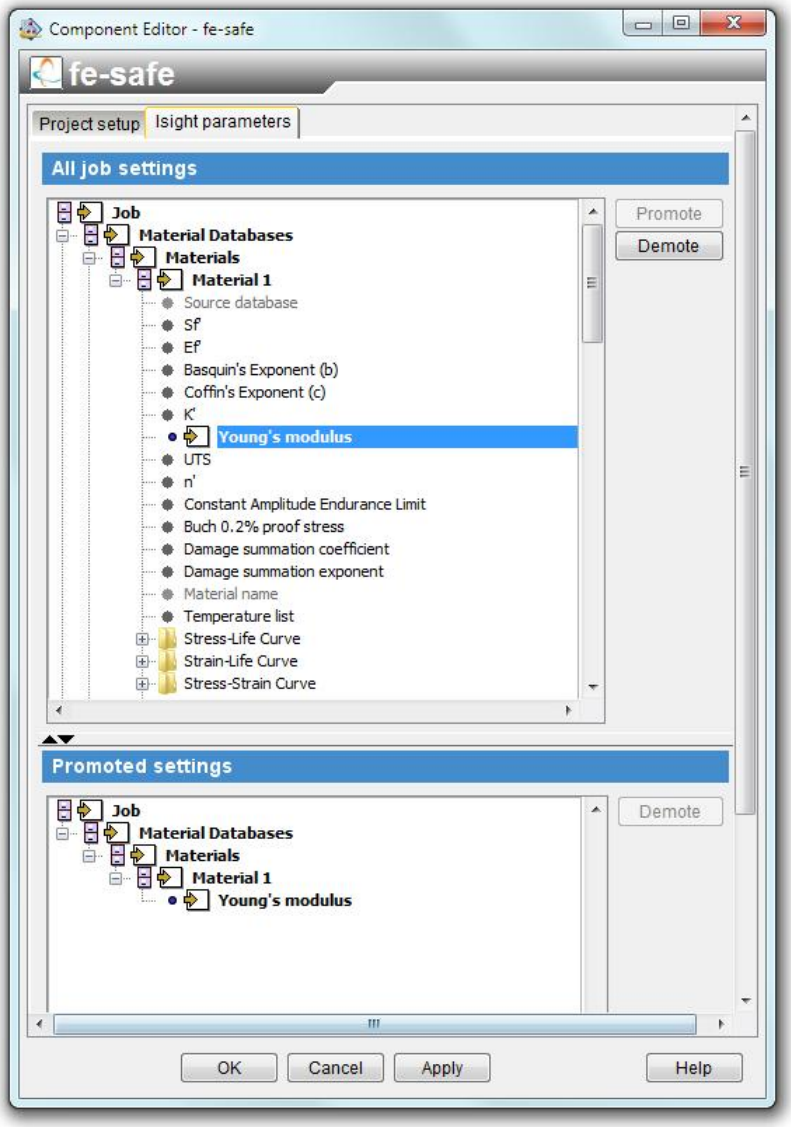

Figure 2-3: The Isight parameters tab in the *fe-safe* component

The input and output solution files are automatically available as parameters and are not presented in the tree. When finished making parameterization choices, click **OK** to commit the changes and return to *Isight Design Gateway*.

The choice you made should be now available as an *Isight* parameter, as shown in Figure 2-4.

Isight is also now aware of what results *fe-safe* is going to export, since these are specified in the project setup. These are presented as output parameters – but their values will not be valid until the component has finished executing. At this point, you can also "wire" up file parameters from upstream that will replace the input file at execution.

***Note**: Some settings, if parametrized, may affect the output contour names (or add/remove contours) between executions of fe-safe. This could mean that **Isight** may not be able to find contour results for that execution, or may find contours it wasn't expecting. This will not cause the component to error, but a warning will be generated in the log.*

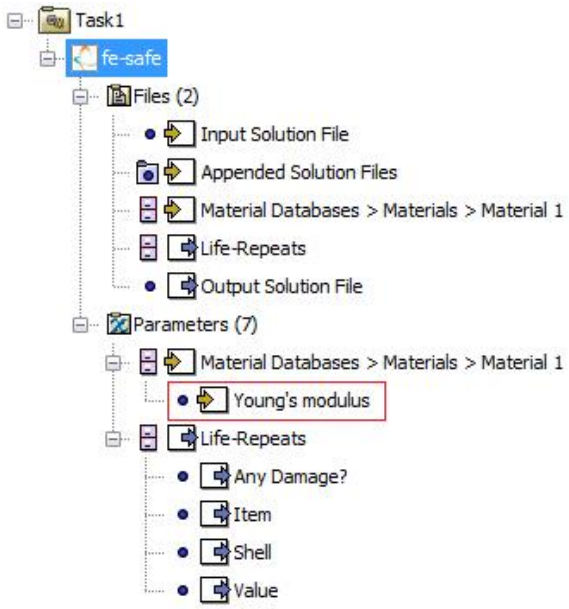

Figure 2-4: Chosen *fe-safe* parameter displayed in Isight

***Note**: On subsequent executions of the model, upstream components will likely update the FEA solution file. If an* fe-safe *system is connected to one of those upstream components, it will refresh its datasets from the most recent FEA solution file that was passed from upstream. If the file name or location is different, the component will respect whatever solution file was most recently supplied. However, the upstream systems must not change the solution file in a way that would invalidate the* fe-safe *project you set up, e.g. by removing required datasets.*

2.3.3 Execution

After executing the component, *fe-safe* will have placed an output file into the working directory, and should have populated all the output parameters with values (unless no damage was reported). The output solution file is available in the file parameter **Output Solution File**. In common with all *Isight* components, the default behaviour is to delete the file after execution, unless it is needed downstream. *Isight* can be configured to keep a copy of the output solution, see *Isight* documentation for details.

You can also see the standard output log from *fe-safe* if you select the **Info** logging level when executing the component. The log will appear in the **Logs** tab in the *Isight Runtime Gateway*.

2.3.4 Limitations

Setting up of custom analysis groups is not supported when using *fe-safe* in a workflow.

Plug-in module settings cannot be read at this release.

At this release, *fe-safe* does not support concurrent execution. Measures have been put in place to stop *fe-safe* being invoked concurrently within an *Isight* model, however, care should be taken not to design an *Isight* model where this could occur.

Some Isight components can be configured to run on a remote system – *fe-safe* is not suitable for this kind of execution due to the large numbers of installation libraries it depends on.

Parametrisation of materials is limited to either using specific material properties or the material names but not both at the same time.

3 Interface Reference

Note that only those interface elements of particular importance for using the *fe-safe* component for Isight are described here.

3.1 *fe-safe* stand-alone and component GUI

When you click **Edit project** in the component dialog box, *fe-safe* opens up in a component mode. The look of this is essentially identical to what you see when you launch *fe-safe* on its own, with some minor differences. These are shown in Figure 3-1.

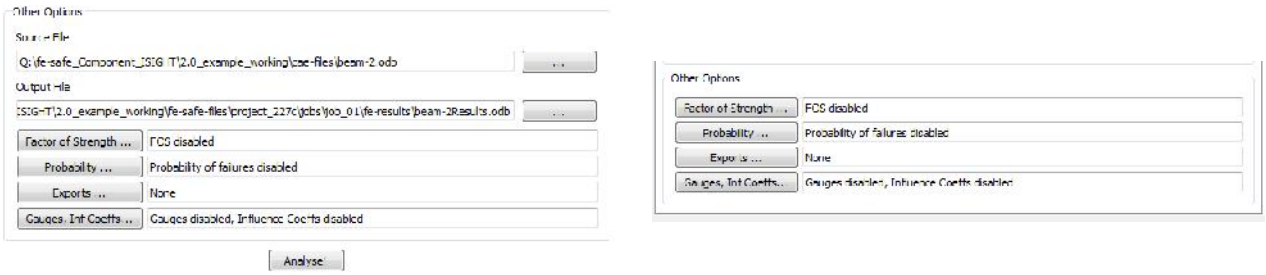

Figure 3-1: *fe-safe* stand-alone GUI (left) and Isight component GUI (right)

The differences are all located in the **Fatigue from FEA** window. In component mode, the buttons for choosing the source file and output file have been removed (they have been moved to the Isight component editor). Also, the **Analyse!** Button has been removed, since the purpose of the Isight component is to run *fe-safe* from inside the simulation process workflow.

3.2 *fe-safe* Project Settings

If you choose to create a new project from the *fe-safe* component in Isight and to then edit the project, it is important to make sure the units in the **Current FE Models** window are configured as expected.

For instance, *fe-safe* defaults the stress units to Pa (as shown in Figure 3-2), those must be changed if required. This is done by right-clicking the top row in the **Current FE Models** window, choosing **Properties** from the context menu and then selecting the stress unit from the drop-down menu.

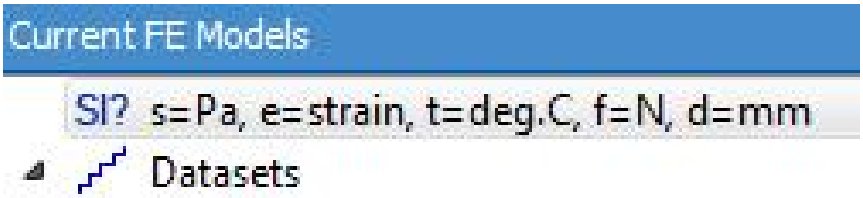

Figure 3-2: Stress unit defaults to [Pa]

3.3 *fe-safe* Component Settings

If you wish to import all increments from an FE analysis (i.e. a sequence of stresses/strains), the relevant box on **Project setup** tab, see Figure 3-3, needs to be deselected.

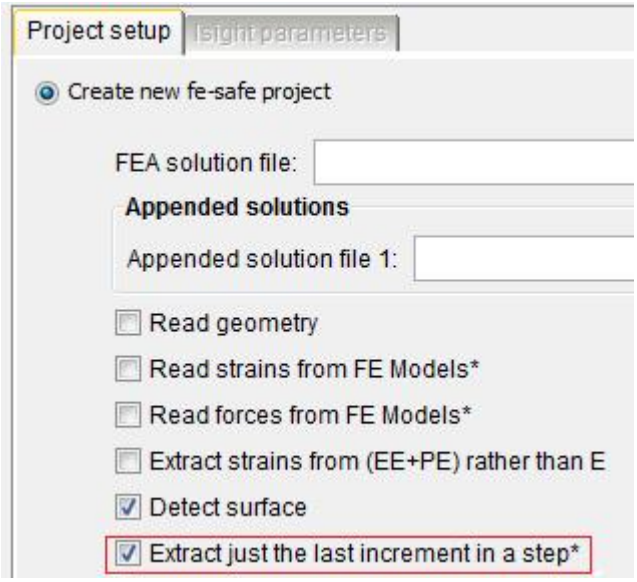

Figure 3-3: Default setting to extract only the last increment in a step

Due to the pre-scan issue mentioned in Chapter 2, Section 2.3.1 you need to be aware of two checkboxes in the *fe-safe* Isight component dialog box. These are **Read Geometry** and **Detect Surface**. If you are using a 3D solid model as input and want to detect the surface, you need to select the **Read Geometry** box. If you are using a 3D shell model, this is not necessary. The checkboxes are highlighted in Figure 3-4.

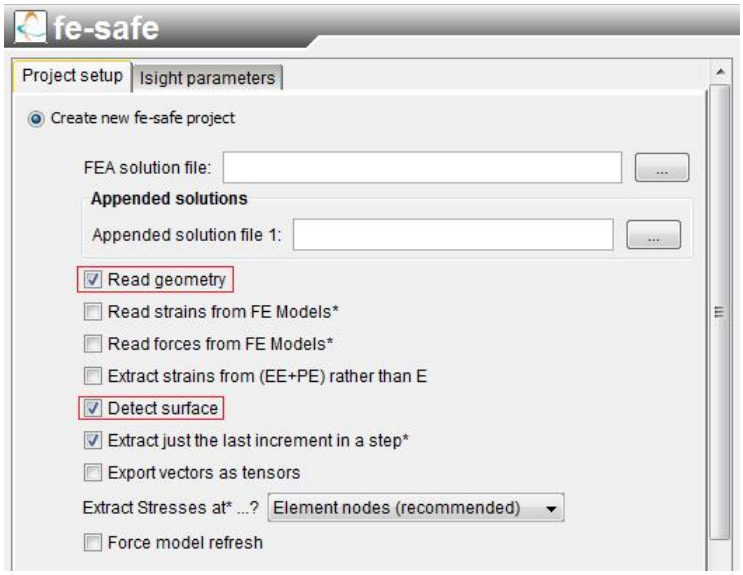

Figure 3-4: Pre-scan issue related checkboxes

4 Preparing FE Models for use with the *fe-safe* component

4.1 Supported Abaqus Releases

Currently the Abaqus releases supported by Isight in *fe-safe* are the same as are supported in the *fe-safe* stand-alone application; this means from Abaqus 6.4 on Windows platforms and Abaqus 6.6 on Linux platforms.

4.2 Model Consideration

There are no additional requirements for FEA modelling and analysis for use in the *fe-safe* component. Any analysis you have performed in the *fe-safe* application can be used with the *fe-safe* component for Isight.

4.3 General Advice

- If you are going to use an existing *fe-safe* project in your Isight simulation process flow, it is recommended to make a backup copy of it.
- Note the required data units to be used [Pa, MPa, N/mm^2...].
- Upgrade your odb if it is older than specified in Section 4.1 of this Chapter.

5 Tutorial A: Standard Fatigue Analysis (Using an odb file)

5.1 Introduction

This tutorial outlines how to perform a standard *fe-safe* fatigue analysis from within Isight. It is assumed that you have experience using *fe-safe*, thus detailed information on how to set up an *fe-safe* analysis are not included in this tutorial. A level of working experience with *Isight* is also assumed, so that the focus of the tutorial is on the workflow rather than configuration details. For a detailed description of Isight, please refer to the *Isight Getting Started Guide* from Simulia.

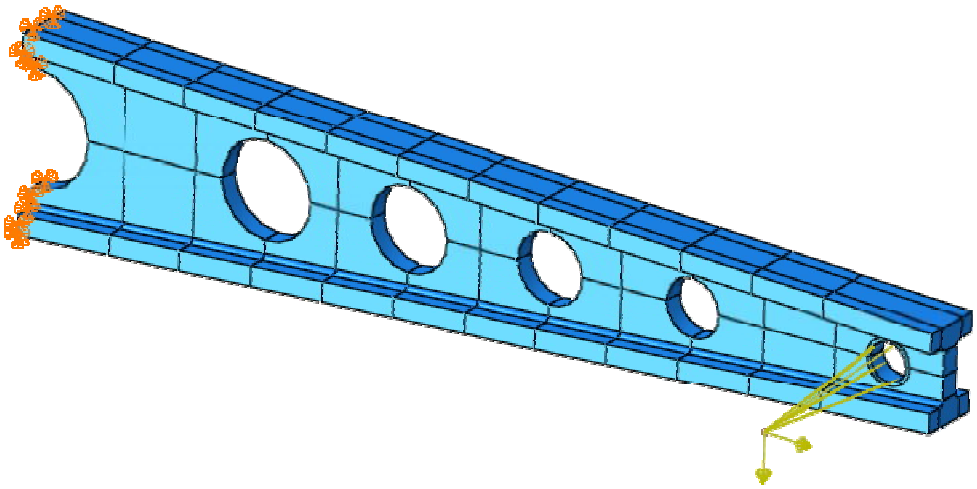

Figure 5-1: Tapered cantilever beam subject to eccentric loading

This analysis is based on a model of a tapered cantilever steel beam subjected to an eccentric load as shown by the arrows, see Figure 5-1. The indicated load is applied in a first general step and then reversed in a second step. The full sequence of stresses (i.e. all increments) are to be used for the fatigue analysis, rather than just the last increment. The load case is defined in Table 5-1 below.

Load Direction	Forward	Reversed
Number of load repeats in one cycle	1	1
Total number of cycles	15000	15000

Table 5-1: Load case definition for the fatigue analysis

5.2 Setting up the Analysis in Isight

5.2.1 Creating the Workflow in Isight

To create the workflow, start *Isight* and make sure you have published the *fe-safe* component as detailed in Chapter 2, Section 2.1. Follow the steps outlined below to complete the *fe-safe* analysis:

1) Drag and drop the *fe-safe* component onto the horizontal arrow in the default flow displayed on the canvas. Your display should look like the one in Figure 5-2.

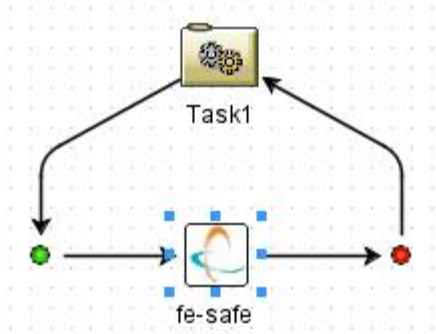

Figure 5-2: The *fe-safe* component in the simulation process flow

2) Double-click the *fe-safe* component to bring up the component editor as shown in Figure 5-3.

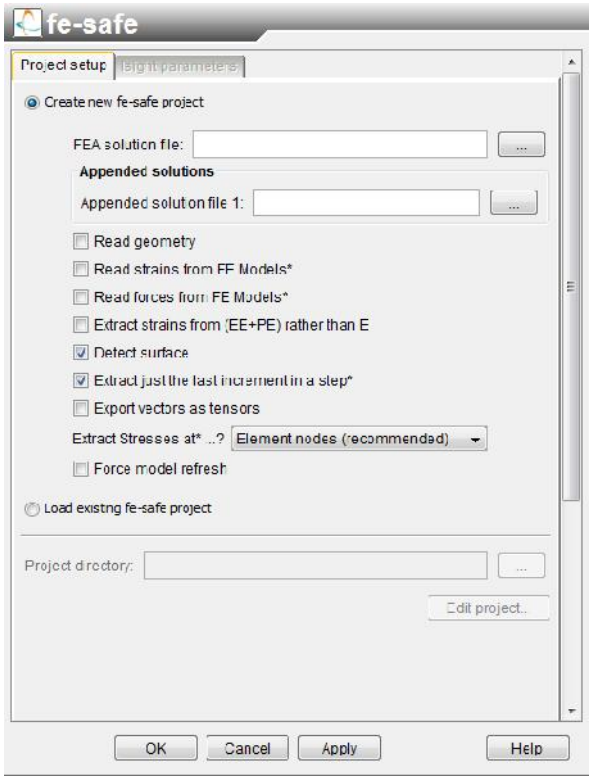

Figure 5-3: The *fe-safe* component in the simulation process flow

3) Click the button next to **FEA solution file:**, navigate to the folder `<DataDir>\Tutorial_A\odb\`, select `cantilever-1.odb` and then click **Open**. The dialog box will look similar to Figure 5-4. Note that the the component automatically suggests a directory in which to save the project.

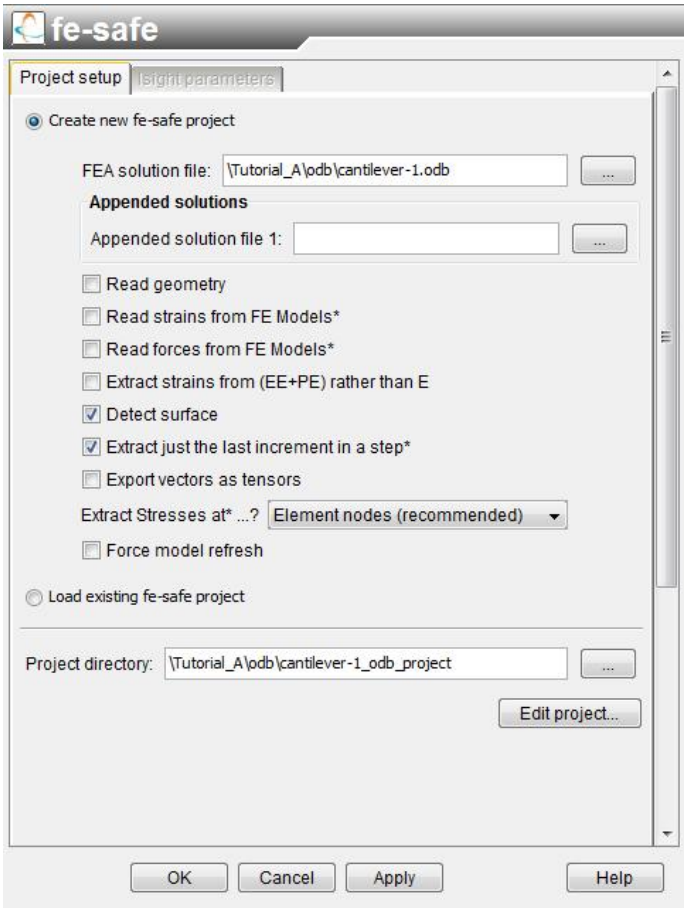

Figure 5-4: The *fe-safe* component with selected solution file and default project directory

At this stage, you will choose a different folder in which to save the project. Further, you will also deselect the option to read only the last increment in a step as all increments of both load steps will be used in the analysis. As a final action before editing the project you will tell *fe-safe* to read the geometry from the 3D solid FEA model.

4) Click the button next to **Project directory:**, navigate to the folder containing the three folders `Isight`, `odb` and `project_tutorial_A` – highlight `project_tutorial_A` and click **Open**. Your dialog box should now resemble that of Figure 5-5.

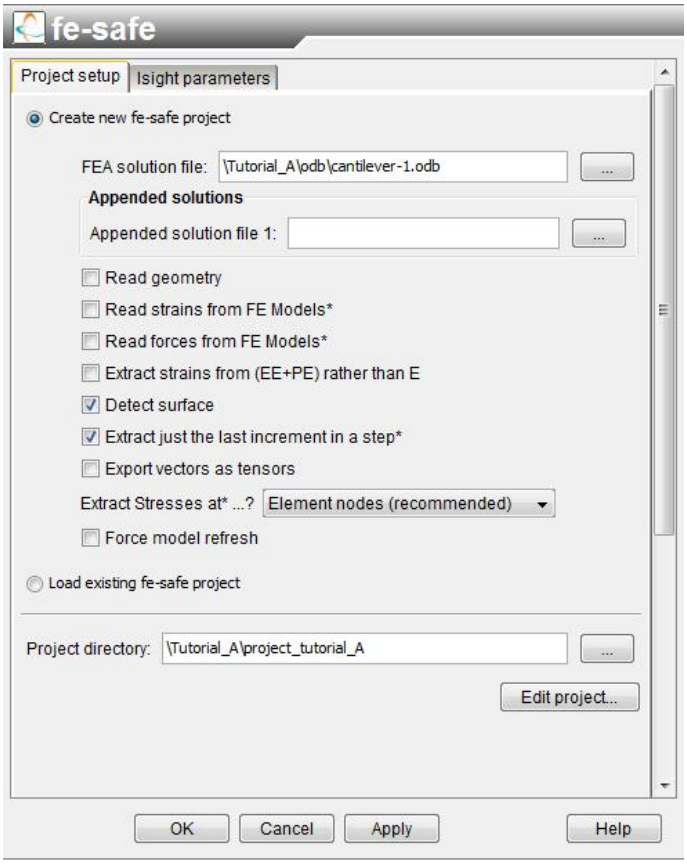

Figure 5-5: The *fe-safe* component with selected solution file and default project directory

5) Deselect the box **Extract just the last increment in a step***.
6) Select the box **Read Geometry**.

Your dialog box should resemble that of Figure 5-6. The component will now:

a. read the file `cantilever-1.odb`
b. read the full geometry
c. detect the model surface
d. read all the increments from the FEA model
e. store the project in the `folder project_tutorial_A`

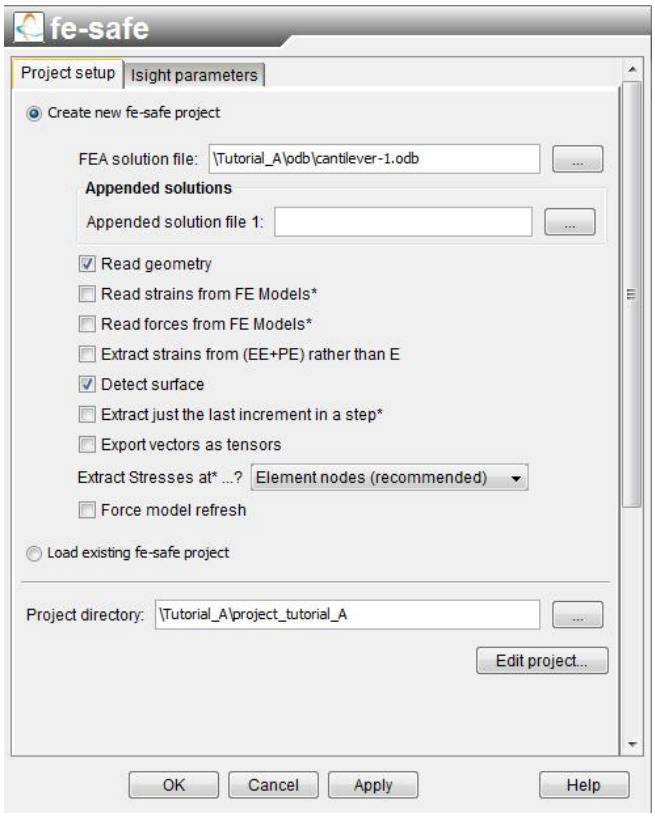

Figure 5-6: The *fe-safe* component with prepared settings before editing the project

5.2.2 Editing the Project Settings in *fe-safe* (component mode)

1) Click the button **Edit Project**... *fe-safe* will open in a component mode.
2) Set the FE model stress unit to **MPa**.
3) Adjust the groups so that only **CANTILEVER_BEAM-1_PICKEDSET26** and **Default** are available.
4) Leave the default surface finish settings.
5) Set **SAE-1144** as material.
6) Define the loading so that it resembles that of Figure 5-7.

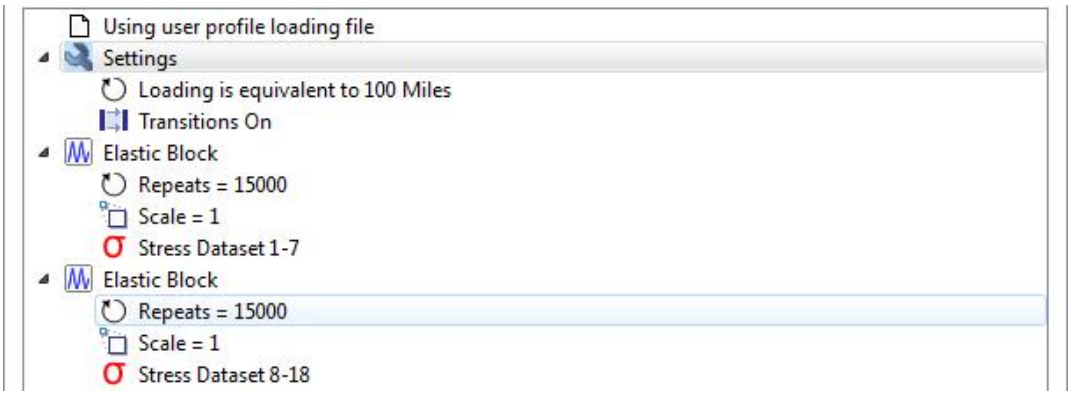

Figure 5-7: Completed load definition in *fe-safe* (component mode)

7) Exit *fe-safe* (component mode) and you will be returned to the *fe-safe* component editor in Isight.
8) Close the component editor dialogue box – the fatigue analysis has now been defined and is ready to run.

5.2.3 Running the analysis and viewing the results

1) Run the analysis by highlighting Task1 and then clicking the run button or by pressing F4.

The analysis runs to completion and presents the following depiction in the Isight Runtime Gateway. The chequered flag indicates a successfully completed task. The figure next to the flag states how many runs have been performed.

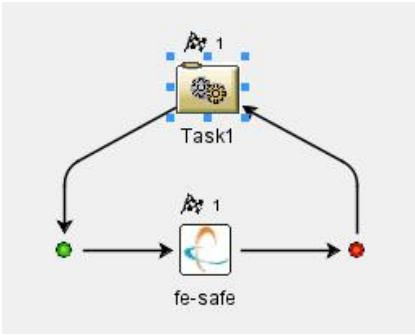

Figure 5-8: The completed *fe-safe* analysis launched from withing Isight

2) Expand the Model Selection tree and highlight the Value output as shown in Figure 5-9. Switch to the history tab to see the life result value in Miles. The value is app. 26.6 miles. Alternatively, you could view the result in the Preview Graph window to the right in the Isight Runtime Gateway.

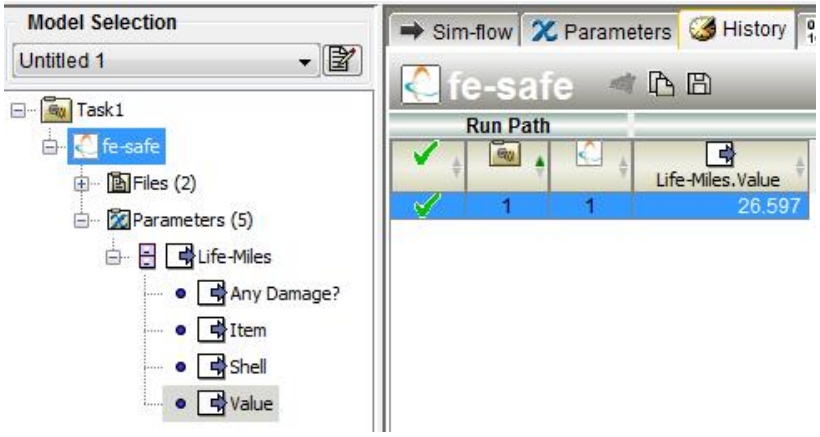

Figure 5-9: Viewing the life result in the history tab

3) Save the results in the folder named `\Isight\` as `single_run.zrf` and then close the Isight Runtime Gateway.
4) Save the simulation process flow as `single_run_flow.zmf` in the same folder.
5) Exit the Isight Design Gateway. You have now completed tutorial A.

6 Tutorial B: Surface Finish Sensitivity Analysis (Using an odb file)

6.1 Introduction

This tutorial outlines how to perform a parametric study of the global surface finish sensitivity on an *fe-safe* fatigue analysis from within Isight.

This analysis is based on a model of a tapered cantilever steel beam subjected to an eccentric load as shown by the arrows, see Figure 5-1, as used in Tutorial A.

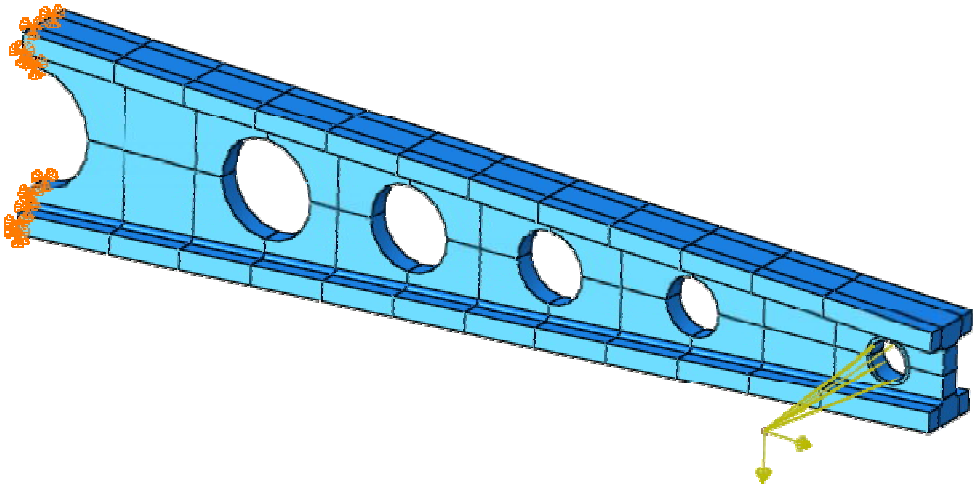

Figure 6-1: Tapered cantilever beam subject to eccentric loading

6.1.1 Creating the Workflow in Isight

To create the workflow, follow the steps as defined in section 5.2.1 with three minor exceptions:

1) The odb file is stored in `<DataDir>\Tutorial_B\odb\` (step 3)
2) Use the file `cantilever-2.odb` instead of `cantilever 1.odb` (step 3)
3) Use `project_tutorial_B` instead of `project_tutorial_A` in (step 4)

6.1.2 Editing the Project Settings in *fe-safe* (component mode)

Follow the steps outlined in section 5.2.2 with one exception (step 4):

- change the **Surface finish** to be defined as a **Value**, and leave that value at **1.0**.

6.1.3 Creating, editing, setting and mapping parameters

1) Back in the *fe-safe* component editor in Isight, select the **Surface finish** parameter from **Groups > Analysis Group 1** and promote it.
2) Close the component editor dialog box.
3) Save your .zmf file.
4) With the *fe-safe* component highlighted, go to the parameters tab. You will see that the surface finish is available as an input parameter (surface description) and the fatigue life is available as an output parameter (Value). Change the surface description parameter **type** from **String** to **Real**, as you will want to pass values for Kt back to *fe-safe* for the subsequent analyses defined in the loop. Your parameters tab should look like Figure 6-3.

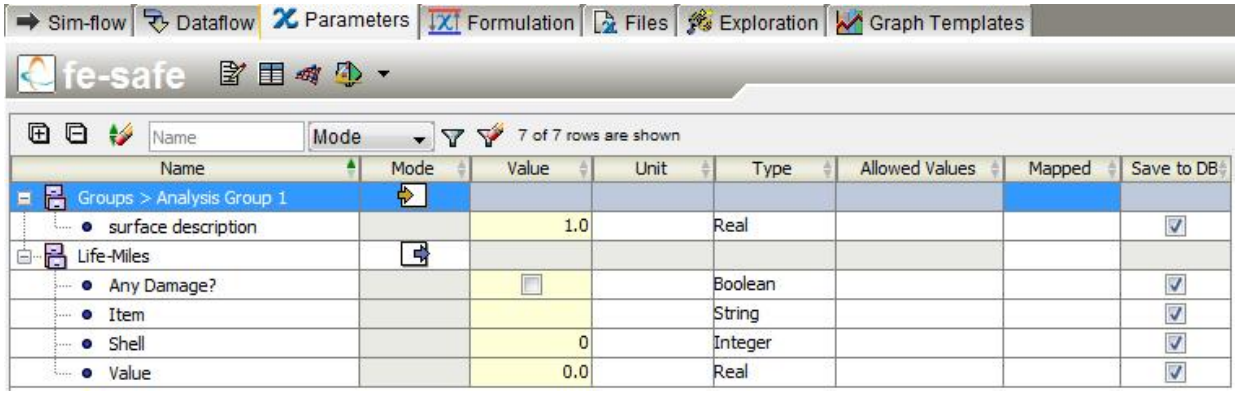

Figure 6-3: Parameter tab settings

5) Go back to the Sim-flow tab, right-click the **Task1** component, choose **Create new** and select **Loop** from the pop-up window. Leave the checkboxes empty and click OK.
6) Highlight the loop, go to the **Parameters** tab and create two new parameters as shown in figure 6-4.

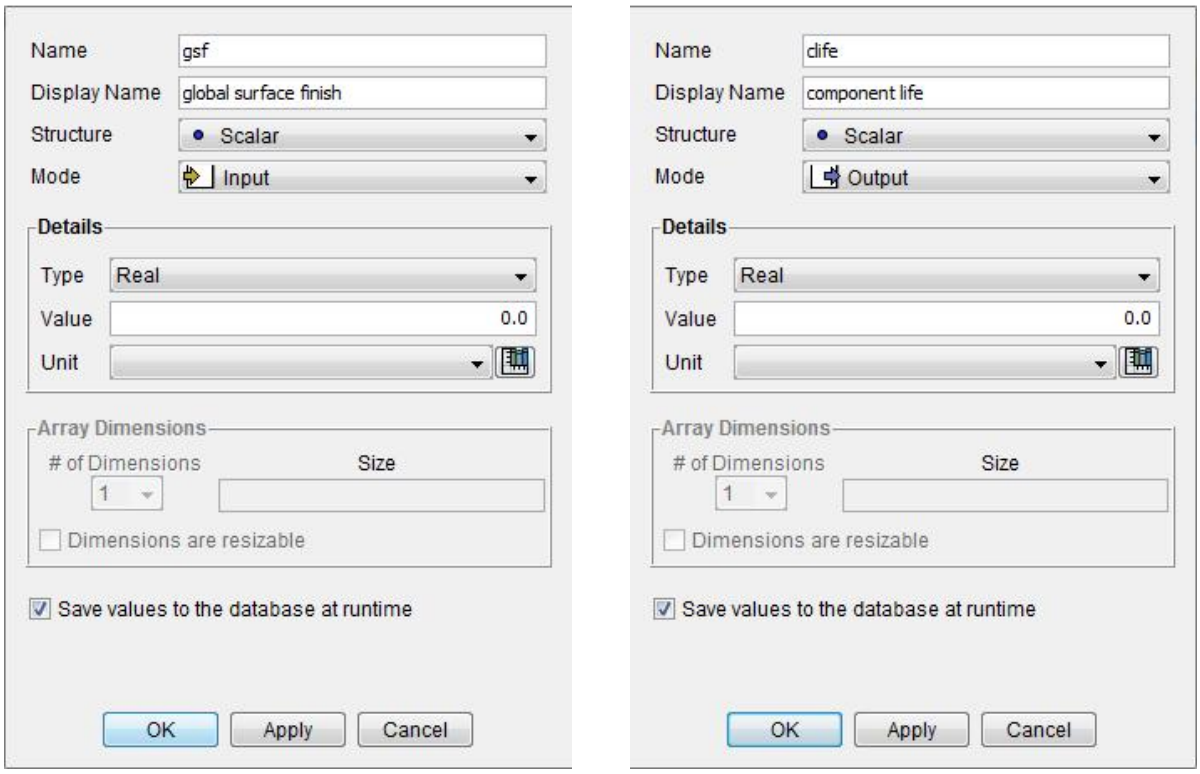

Figure 6-4: Creating the two parameters

7) The next step is to map the components between the loop and *fe-safe*. Keep the loop highlighted in the Sim-flow tab, go to the Dataflow tab and map the parameters according to Figure 6-5.

Figure 6-5: Parameter mapping

6.1.4 Configuring and running the loop

1) Go back to the Sim-flow tab, double-click the loop component and set up the loop according to Figure 6-6.

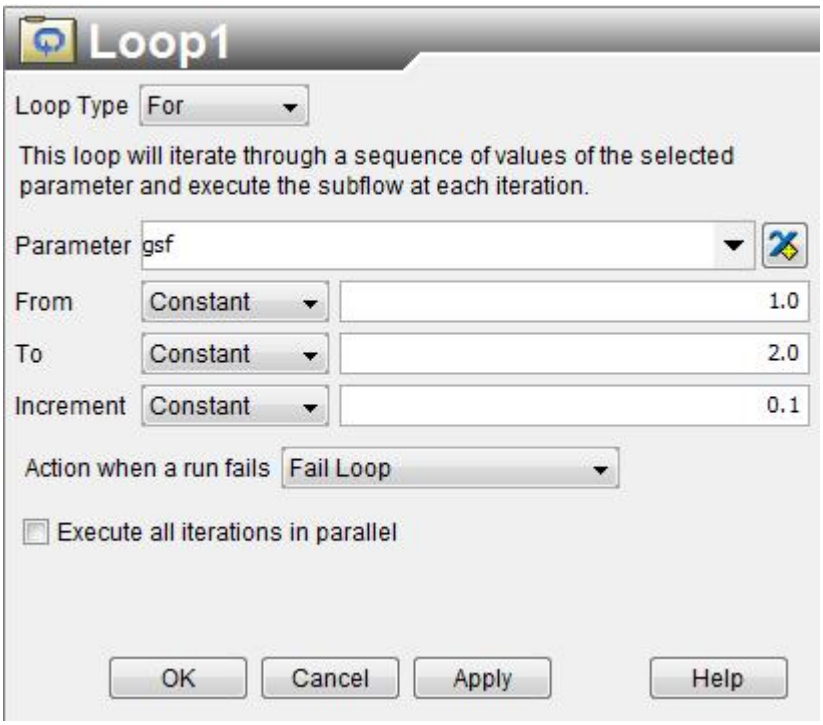

Figure 6-6: Loop setup

2) Start the loop by clicking the run button or by pressing F4. The Isight Runtime Gateway opens and you can follow the progress in the different tabs.

6.1.5 Monitoring the analysis status and viewing the results

1) Expand the **Parameters (3)** icon in the **Model Selection** tree.
2) Switch to the **History** tab. You will see the results of the loop runs and you can watch them update in the **Preview Graph** window to the right, as shown in Figure 6-7.

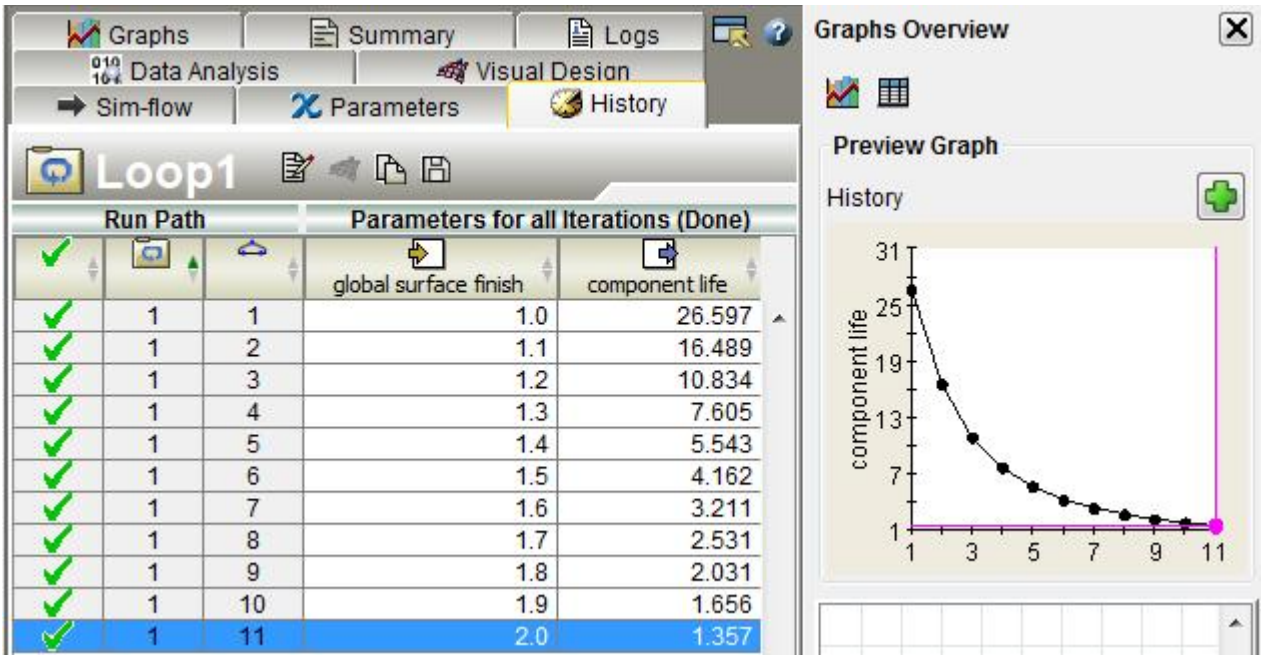

Figure 6-7: Following the results progression

When the analysis has completed, the Sim-flow will appear as in Figure 6-8. It shows that one loop (sensitivity analysis) has been performed, with 11 *fe-safe* analysis runs successfully completed.

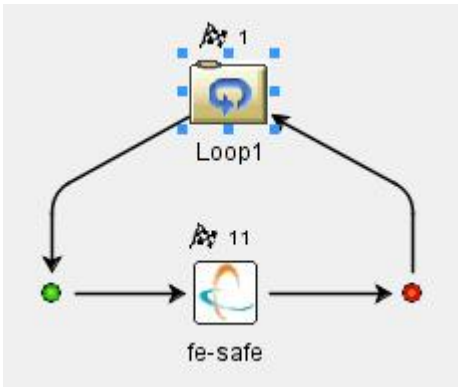

Figure 6-8: The successfully completed sensitivity analysis

You will now manipulate the results plot and briefly interpret the message from a Life-Kt-Diagram.

6.1.6 Plotting and interpreting the results

1) Create a line plot from the Graphs tab to resemble that of Figure 6-9.

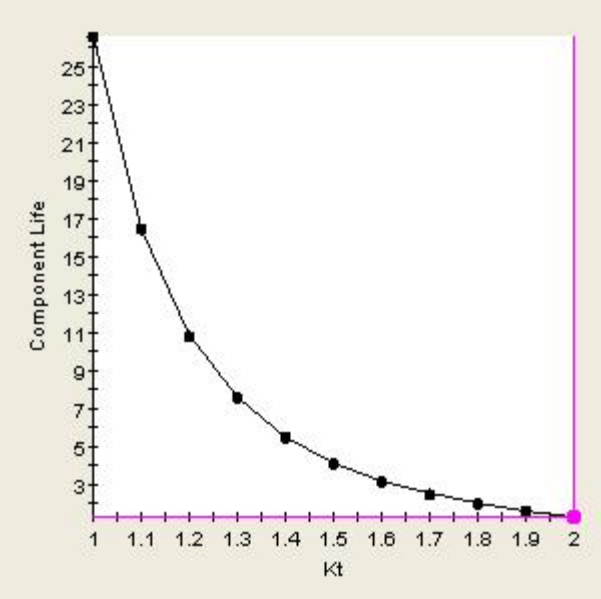

Figure 6-9: The Life-Surface Finish diagram

The diagram shows that the sensitivity of the component is most severe between Kt values of 1.0 – 1.3. After that, the sensitivity is significantly lower, and after a Kt value of 1.6 it is almost negligible, relatively speaking. This kind of diagram can be used in two ways:

I. To find out how much the surface needs to be improved (process control) in order for the component to meet the criterion of a life specification.
II. To find out how much the surface finish can be lowered whilst still retaining the required component life. This could be used in cost reduction and component over-design analysis.

2) Save the results as `loop_run_flow.zrf` in the `\Isight\` folder. Close the Runtime Gateway.
3) Save the process flow, then close the Design Gateway.
4) Exit the Isight Design Gateway. You have now completed tutorial B.